International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 10, October- 2014

Finite Element Modeling and Analysis of CFRP Composite Stiffened Panels for Post Buckling Behavior

Krishna Mouly Vasireddy
P. G. student, Aerospace Engineering
MLR Institute of Technology
Hyderabad, India

Madhavi Nagi Reddy
Asst. Professor, Department of Aerospace Engineering
MLR Institute of Technology
Hyderabad, India

Dr. S. Srinivasa Prasad
Professor & Head,
Department of Aeronautical Engineering
MLR Institute of Technology
Hyderabad, India

Abstract— **Stiffened composite panels are extensively used in aircraft structures because of their high stiffness to weight and high strength to weight ratios. Stiffened panels prone to buckle under in-plane loads and understanding the load carrying capacity beyond buckling is very important. The load carrying capacity of the panels depends on the post buckling behavior and strength. If the panels possess significant post buckling strength then the structure can be allowed to buckle before the design load and this will significantly reduce the cost and weight of the structures. The aim of the project is to analyze the buckling, post-buckling behavior and failure characteristics of simply supported stiffened panels under uni-axial compressive load. Finite Element Analysis (FEA) is a very good computational technique that can be used for the study. These FE results obtained can be verified with the help of experimental results.**

Keywords — Stiffened panels; Buckling; Post-Buckling; ANSYS.

I. INTRODUCTION

Laminated composites are gaining wider use in mechanical and aerospace applications due to their high specific stiffness, specific strength and lower weight. In addition to the higher strength and stiffness, in a fibrous composite, the mechanical properties can be varied as required by suitably orienting the fibers [1]. When slender structural members are subjected to axial compressive load, they instead of failing in compression they may buckle due to lower bending stiffness compared to that of axial or membrane stiffness. If the axial load is further incremented the lateral deflection would increase rapidly near buckling load and eventually the member would collapse [2]. Linear finite element analysis has been adopted by designers as the standard tool for the structural analysis of high performance marine and aerospace structures. However, it is known that the geometrically non-linear behavior of laterally loaded FRP plates is significant, even at low load levels [3].

Current aerospace industrial practices of CFRP composites are to design the panel to buckle only at the ultimate load and its post-buckling strength is not considered for the design at all. If the panel is allowed to buckle between the limit load and the ultimate load, the post-buckling strength will get utilized to improve the design by reducing the weight of the panel and at the same time the cost.

The objective of the project is to study buckling and the post-buckling behavior of CFRP stiffened panels using finite element methods and validate with the available experimental results in the literature [1]. An equivalent model of the original stiffened panel is used to study the post-buckling behavior and the panel failure. To study the failure of the panel, Tsai Wu strength index and Maximum stress failure criterions are used. Buckling and post buckling analysis of the CFRP stiffened panels using three finite element models, SHELL181 (4-noded), SOLID185 (8-noded) and SOLSH190 (8-noded) using ANSYS has been studied. The load-displacement plots, bending strain plots are plotted and subsequently compared with the available experimental results.

II. GEOMETRY MODELING OF THE CFRP STIFFENED PANELS

Equivalent Flange (EF) model

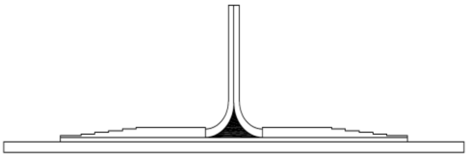

Figure -1 Actual flange model used in Aircraft skin stringer

In any vehicle stringer are structural components which are used to support the skin. The aircraft skin and stringer interface is shown in figure -1. The major issue for analysis was modeling the triangular portion formed at the skin-flange interface junction, which consists of UD fiber bundles. The finite element modeling of stiffened panels can be modeled using SOLID elements where the triangular portion can be modeled using combinations of brick and wedge shaped solid elements. But when it comes to the shell, it is quite a complicated matter. So, model has been simplified with the triangular portion formed at the skin-flange-web junction by connecting web of the stiffener at right angle to the flange of the stiffener as shown in figure – 2.

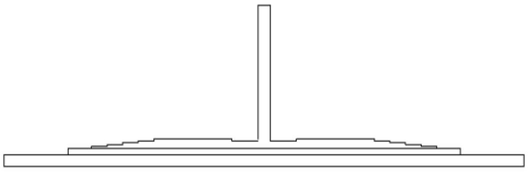

Figure - 2 Simplification of original geometry model to an equivalent flange model

The constraint equations are imposed on the nodes of the skin elements and the flange elements in order to enforce compatibility along the interface. The displacement field according to the Mindlin plate theory is as follows

$U_X = U_X^0 + Z\,\Psi_x$
$U_Y = U_Y^0 + Z\,\Psi_y$
$U_Z = U_Z^0$

Where

U_X, U_Y, U_Z - displacements of plate elements
U_X^0, U_Y^0, U_Z^0 - the displacements at mid-surface of the plate elements
Ψ_x , Ψ_Y - the rotational degrees of freedom of the plate elements
Z - distance of extreme fiber to mid-surface of the plate elements.

The flange and skin elements must be tied together in both displacement and rotations along the interface. When shell elements are used, the skin and web elements of the panel have a thickness of 2.4 mm while the flange element is of 1.8 mm thick. So care must be taken while applying the uniformly distributed load across the loaded edge. In the node offset method for the shell element, for the plate 1N/m load was applied and 0.5N/m is applied to the flange which is 1.2 mm thick in the absence of intermediate and capping layers and 0.75N/m is applied to the flange which is 1.8 mm thick in the presence of the intermediate and capping layers. Thus a uniformly distributed load is applied throughout the edge.

III. FINITE ELEMENT MODELLING AND ANALYSIS

A. *Material Modeling*

The material used for making the panel is T300/914 carbon epoxy pre-preg with co-cured, co-bonded fabrication process using vacuum bag high temperature autoclave curing. The 3D orthotropic properties are as follows:

Elastic Modulus	E_{11} = 130 GPa	E_{22} = 10 GPa	E_{33} = 10 GPa
Poisson's ratio	μ_{12} = 0.35	μ_{23} = 0.44	μ_{31} = 0.35
Modulus of Rigidity	G_{12} = 5 GPa	G_{23} = 4.1 GPa	G_{31} = 5 GPa

Table – 1 Material properties of CFRP

The lay-up for skin as well as web is $[\pm45/0/45/0/-45/0/90]_s$ and while that of a flange is$[90/0/-45/0/45/0/\pm45]$. Two intermediate layers are ±45 layers between the skin and stiffener flanges and additionally 2 capping layers on top of the each flange section (±45) are present in the panel to add additional stiffness to the panel. All layers are 0.15mm thick.

Element selection

The equivalent flange (EF) stiffened panel is modeled using SOLID185 element. SOLID185 is an 8 node element which can be used to model layered structures. The element has 3 degrees of freedom i.e. translations in x,y and z directions with the capability of large deflection and stress stiffening. Along with these elements we have to use shell 181 as well.

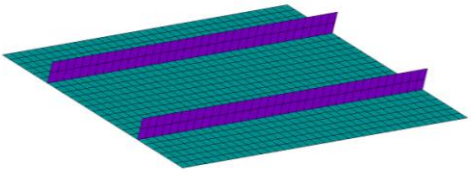

Figure - 3 Stiffened panel modeled using shell element with node offset method

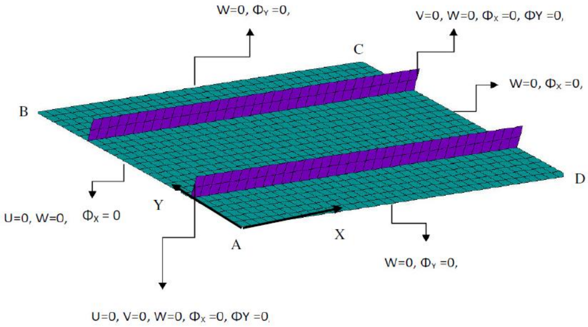

Figure – 4 FE model with boundary conditions

IV. RESULTS AND DISCUSSION

The displacements in Z-direction for the stiffened panel for 4 modes have been extracted. The buckling load after addition of these layers has been predicted to be 99.941KN, which is in good agreement with the experimental result. So this explains that these layers contribute significantly to the stiffness of the panel.

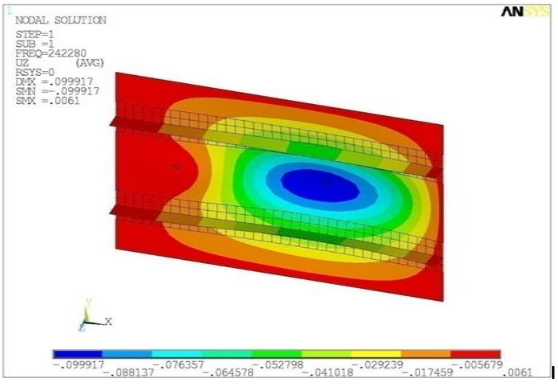

Figure – 5 Buckling mode shape - 1

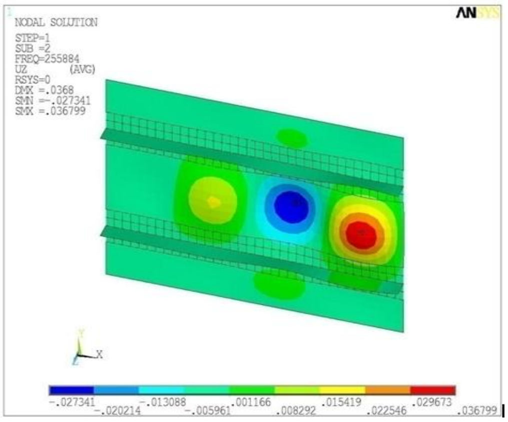

Figure – 6 Buckling mode shape – 2

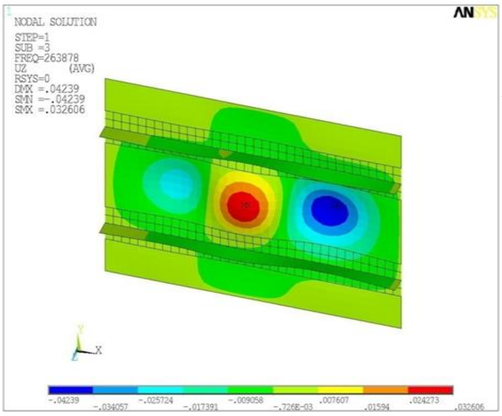

Figure – 7 Buckling mode shape - 3

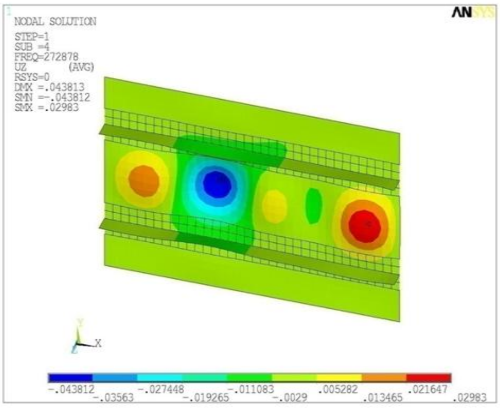

Figure – 8 Buckling mode shape - 4

Figures 5 through Figure 8 shows the buckling mode shapes of the EF with intermediate and capping layers. Two failure criterions are used to find out the failure of plies in the composite structure. With the given material strength properties, Tsai- Wu and Maximum stress criterions has been used to predict the ply failure and failure progression.

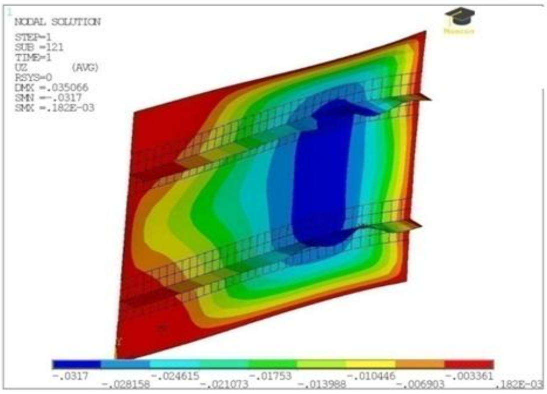

Figure – 9 Failure criterion in Z direction

Based on the Tsai Wu strength indices, the first ply failure is found to occur at 135KN in the 45 degree plies of top and bottom surface of the skin close to the loaded edge. Further increasing the load, failure of other 45 degree plies is also observed in the same region. Later all the plies in the marked region, shown in, failed with increase in the load. As the load is increased, web had undergone compression, especially at the free edge. Failure of 0 degree plies has been observed in the web region. The Web showed failure at 145KN with 0 degree plies failing at the free edge

V. CONCLUSION

Based on the results that we got from the analysis the following conclusions have been drawn:

- The Composite cylinder designed is meeting the stipulated internal pressure. The procedure followed has worked out to be efficient in accurately predicting the structural response of composite components.
- It meets all the desired functional requirements.

VI. REFERENCES

[1] Jonathan E. Rich, “Design Optimization Procedure for Monocoque Composite Cylinder Structures Using Response Surface Techniques,” September 12, 1997 Blacksburg, VA

[2] Jin Guang Teng,”Buckling of thin shells: Recent advances and trends” American Society of Mechanical Engineers, 1996

[3] Reissner. E.,“On the theory of bending of elastic plates,” J. Math. Phys.,23, 1944.

[4] John. T. Dorsey, “Structural analysis of the space shuttle solid rocket booster attached ring,” NASA Technical Memorandum 100510, 1988.